

Easy-PC

V23.0 Supplement

Copyright © 1998-2019 WestDev Ltd. All rights Reserved. E & O E

Number One Systems

20 Miller Court
Severn Drive
Tewkesbury Business Park
Tewkesbury
Glos GL20 8DN
United Kingdom

Phone: 01684 296 501

Email: sales@numberone.com

Technical: 01480 382 538

Email: support@numberone.com

Web site: www.numberone.com

The product is licensed by WestDev Ltd to the original purchaser of the product for use only on the terms set forth in the WestDev End User Licence Agreement. Copying, duplicating, selling or using the product contrary to these terms is strictly forbidden.

This Manual Copyright to WestDev Ltd. All Rights reserved. No part of it may be copied, photocopied, reproduced, translated, or reduced to any electronic medium or machine-readable form without WestDev Ltd. prior written permission.

Information in this manual is subject to change without notice and does not represent a commitment on the part of the vendor.

Number One Systems, Easy-PC, Easy-Spice, Easy-Router, ProRouter, Pulsar, Layan, Analyser, MultiRouter, Filtech, Z-Match, StockIt and all variants thereof are Trademarks of WestDev Ltd.

All other trademarks acknowledged to their rightful owners.

Number One Systems, a trading division of WestDev Ltd.

While every care has been taken in the preparation of this manual WestDev Ltd cannot be held responsible for any errors or omissions within it. If informed we will add comments and features which you may like to see written which may help others using this manual. Please send your comments through the technical support desk.

Manual date: 28-August-2019 Issue 2

Contents

CONTENTS.....	3
CHAPTER 1. GETTING STARTED.....	5
Installation	5
Running Easy-PC 23.0.....	6
CHAPTER 2. NEW FEATURES IN EASY-PC V23.....	7
Introduction.....	7
No Connect Pins.....	7
No Connect and Update Component.....	8
No-Connect Pins in your Design	8
No Connect Pin Rules and Checking	8
No Connect Pins and Properties	9
No Connect Pins and Display	9
No Connect Pins and Reports.....	10
New Warnings Page in Preferences.....	11
Select Mask.....	11
Open Hyperlink.....	12
Refresh Symbol in Component Editor	13
Wizards for New Schematic and PCB Designs	13
New Schematic Wizard.....	14
New PCB Wizard	14
Configuration Files	16
Save to Library Displays Information About Duplicates.....	17
Add Component and Library Manager Dialogs Resizable	18
Export Images as Bitmap and Metafile	18
Add Text Choose Style and Alignment.....	18
Add Track or Shape Off-grid	19
Highlight Net Interactive Mode.....	19
Make Cutouts from Shapes	20
Group Name Shown On Status Bar	21
Display Grid in Previews	22
Cross Probe Single-shot Operation	24
BOM Composer turn off Auto-generate Report	24
Plotting Auto-load Job File	25
Display Net Name on Component Select (SCM).....	25
Move Net Name Highlights Net (SCM).....	26
Dangling Connections Report (SCM).....	26
Height Check Areas Renamed to Component Areas (PCB).....	26
Component Rules (PCB).....	26
Auto-place Using Rules (PCB).....	28
Add Track Action on Cancel (PCB)	29
Edit Package in Library (PCB).....	29
Enhanced Functionality for Create Fanout Pattern (PCB)	29
Online Design Rule Check for Components (PCB).....	30
Delete Unused Net Match (PCB).....	30
Add to Net on Pour Area (PCB).....	30
Pour Copper adds DRC Error Markers (PCB).....	31
Rename Component “immediate” mode (PCB).....	31
Swap Components (PCB)	32
Intelligent Gerber Import Improvements (PCB).....	32

Chapter 1. Getting Started

Installation

Backing up your files

If you already have Easy-PC installed, please remember to back up all your libraries, Technology files and any other data files before proceeding with the installation of the new version. The installer should not overwrite any of your own named files, but it can re-install new copies of our standard data files so if you have changed any of those files it is important to back them up first. If you are uncertain, check the time/date stamp on the file but in any case, make a back-up.

Of course, backing up your data is important not only for the upgrade but also at regular intervals during design.

Installation from a download link

If you have been provided with download link, use this to download the installation executable, named EasyPC.exe. Find it in your *Downloads* folder using Windows Explorer and double-click it. You'll need to type (or copy/paste) the password provided to unpack this file. Once the unpack password has been successful, you will be allowed to continue with the installation.

All other instructions should be followed until you click **Finish** to complete the installation.

The installation is the same for new and existing users alike. Existing users with versions prior to this latest version can install the new software over an existing installation without deleting the old one first.

Installation From CD

Installation is via the *autorun* setup. Insert the CD-ROM into your CD-ROM drive and wait a short time. The CD-ROM will run up to speed and an Easy-PC Welcome screen will appear. If *autorun* has been disabled on your computer you must execute the 'setup.exe' program using the **Start** menu and **Run** command from the Windows task bar.

With the installer running, once the **Welcome** screen is displayed, double-click on the **Install Easy-PC - Version 23.0** option, or click then press **Run**. Following the instructions on the screen, you should use the same **Destination Folder** for the Program Files as your existing Easy-PC program files.

All other instructions should be followed until you click **Finish** to complete the installation.

Note: CDs are no longer supplied by default. A download link would have been supplied unless otherwise requested.

Installing over existing Easy-PC software

If you already have an earlier version of Easy-PC installed on your system and you wish to install the new version into the same folder as the earlier one, please note that you will then end up with both versions listed in the Windows Control Panel list of installed applications.

If you don't want the earlier one to be listed in the **Control Panel**, you will need to un-install that version **before** you install the new one. If you install the new software into the same folder as the old version then try to un-install the old one, you will find that the new software will not run as the un-install will have removed many or all of the program files.

If you wish to install and use the new version without removing the old one, you will need to install the new version into a different folder. The two versions will then operate independently and either can be un-installed without preventing the other from running.

Data Files Location

There is a step in the **Setup** installation wizard that asks you where you want to place data files (for example, Libraries, Technology files, etc). The default is always to use the common documents folder, “Users\Public\Documents\Easy-PC” on Windows 7, 8 or 10 (or the local language equivalents) if you are installing for All Users, or into your own Documents folder if installing for current user only.

Running Easy-PC 23.0

As with all **Easy-PC** programs, an icon will appear in the **Number One Systems** folder, or the Start pane if using Windows 10, you may also wish to create an **Easy-PC** Shortcut icon on your desktop.

To start the program, double-click on the **Easy-PC** icon from the **Number One Systems** folder or the **Start** pane if using Windows 10.

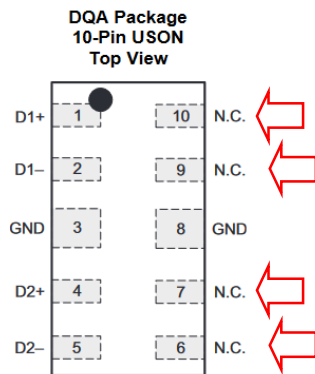
Chapter 2. New Features in Easy-PC V23

Introduction

All features are categorised as being applicable to Schematic (SCM), PCB, or more general use (unmarked).

No Connect Pins

Some devices have pins that have no internal connection within the component and need to be identified so that they are not to be connected. For example, the ESD protection diode package from Texas Instruments has four such pins:



To help ensure that these are not accidentally connected in your design, a pin in a Component can now be marked as **No Connect**. This status is applicable to both Schematic and PCB designs.

This property is accessed via the **Pin Assignments** grid of the **Component Editor**. A new column **No Connect** allows to toggle the No-Connect state of each pin. A display of **Yes** in this column indicates the pin cannot be connected to.

Gate	Sch Symbol	Sch Symbol	Sch Terminal	Pcb Symbol	Component Pin	Net (Class)	No Connect
Name	Name	Terminal Name	Number	Pad Number	Name/Number	Name	
a	93C66BX-X_C	D1+	1	1	1		
		D1-	2	2	2		
		GND	3	3,8	3,8		
		D2+	4	4	4		
		D2-	5	5	5		
		NC	6	6,7,9,10	6,7,9,10		Yes

To use this feature in your design, the No Connect status has to be applied to your component in the **Component Editor** first. It is then added to the design (using **Add Component**) or to an existing component in the design (using **Update Component**).

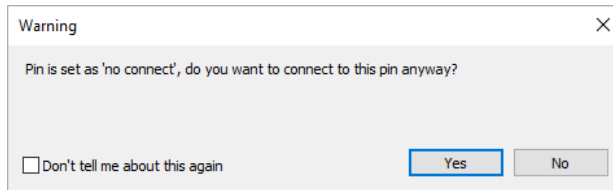
Note, the No Connect status is associated with each package defined for a component, the same as Component Pin Names. Ensure that each package that requires it has the status applied.

No Connect and Update Component

When using **Update Component** to bring in changes made to a component, for example the No Connect status, none of the other switches for **Existing Component Values** or **Positions** affect the NC status.

No-Connect Pins in your Design

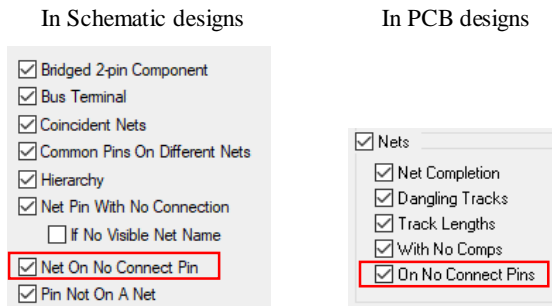
If you attempt to start or end a connection or track, or use the **Add To Net** command, on a pin marked as **No Connect**, the application will display a warning dialog asking you whether you really want to connect that pin.



This is not like a Spacing rule that is strictly enforced, it is there for your guidance, so you can still connect the pin if you really need to do so in a particular situation. Sometimes for example it could be preferable to tie NC pins to ground.

No Connect Pin Rules and Checking

Design Rule Checking in both Schematic and PCB editors have a new check for No Connect pins having nets assigned, to assist in applying this property retrospectively to existing designs by editing components and updating them in the design.



Translate to PCB and **Integrity Check** will also report warnings about **No Connect** pins in the Schematic that have nets assigned to them:

```
Transfer from Schematic to PCB.
-----
Report Written:      Thursday, July 04, 2019
Design Path:        C:\Designs\Route66\
Design Title:       Route66_02
Created:            04/07/2019 04:03:53 PM
Last Saved:
```

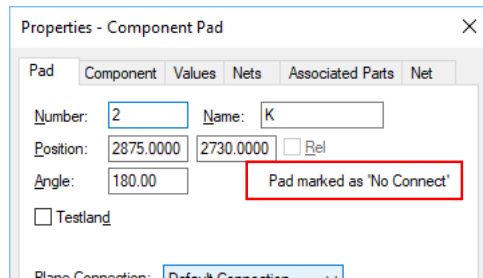

Editing Time: 10 min
 Units: thou (precision 2)

Warning: connected 'No Connect' pin 2 on D1

End Of Report.

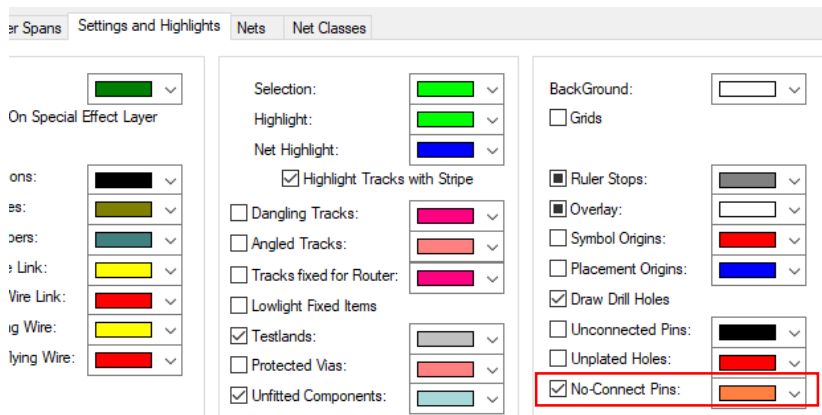
No Connect Pins and Properties

Once defined as NC in the Component, the **Properties** of a selected **Component Pad** will display a note on the dialog to show that the **Pad is marked as No Connect**:



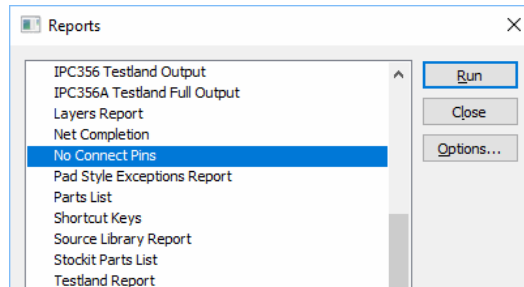
No Connect Pins and Display

The **Display** dialog has a new colour setting in **Settings and Highlights** that allows you to easily visualise the **No Connect Pins** in your Schematic or PCB designs:

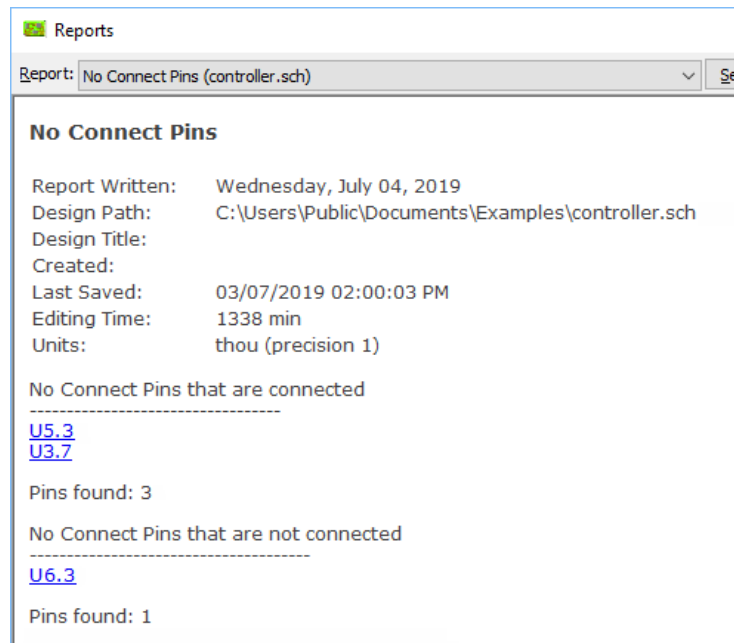


No Connect Pins and Reports

The standard **Reports** option on the **Output** menu now has a No Connect report that shows No Connect pin connected to a net and ones that have this status but are not connected.



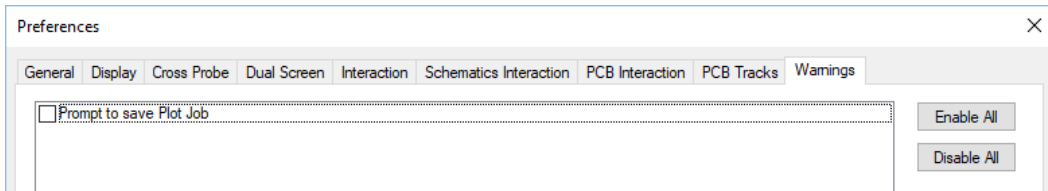
The report prints two separate blocks of data for NC connected and NC not connected:



New Warnings Page in Preferences

Within the **Preferences** dialog, there is now a **Warning** page.

Some messages currently being displayed by the application as normal yes/no (or OK/Cancel) message boxes have been changed to use the warning dialog which includes the option for “don’t tell me again”. These include deleting fixed items, deleting the copper belonging to a pour area being deleted, and creating a new named style. It will only be populated if they have been switched off and may contain nothing in the list on a new installation.



Select Mask

The new **Select Mask** capability provides you with a quick and easy method of filtering the selection for only certain types of items when working in **Select Mode**.

There are four new buttons on the **Edit** toolbar and corresponding entries on the **View | Select Mask** menu and on the shortcut menu in **Select Mode**. These provide a simple method of telling Select Mode whether to pick or ignore items based on four categories:



Components (component shapes and pads but not component names)



Routing (including Vias) in PCB and Connections in SCM



Shapes



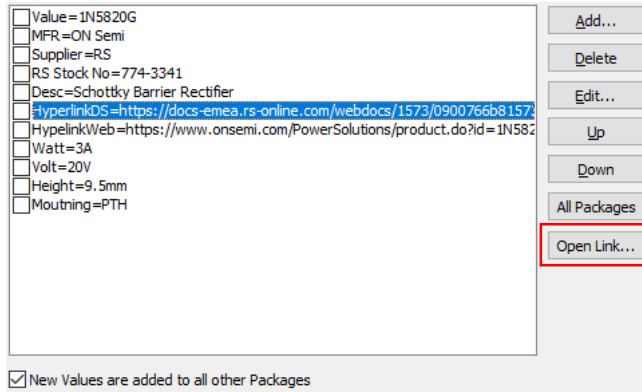
Text

The default state is to have all four options enabled, you can then toggle them off and on from the toolbar buttons or menu commands when you need to.

Note: The **Select Mask** settings do not apply in modes other than **Select Mode**, as these modes already have a restricted set of selection settings appropriate to the particular mode or command.

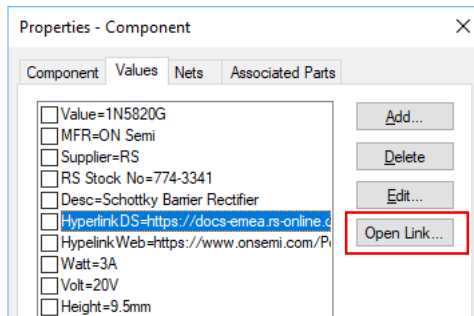
Open Hyperlink

On the **Values** page of the **Component Editor**, a new button **Open Link** will open the selected value item as a hyperlink for any value that has text that looks like a hyperlink or a path to a file.



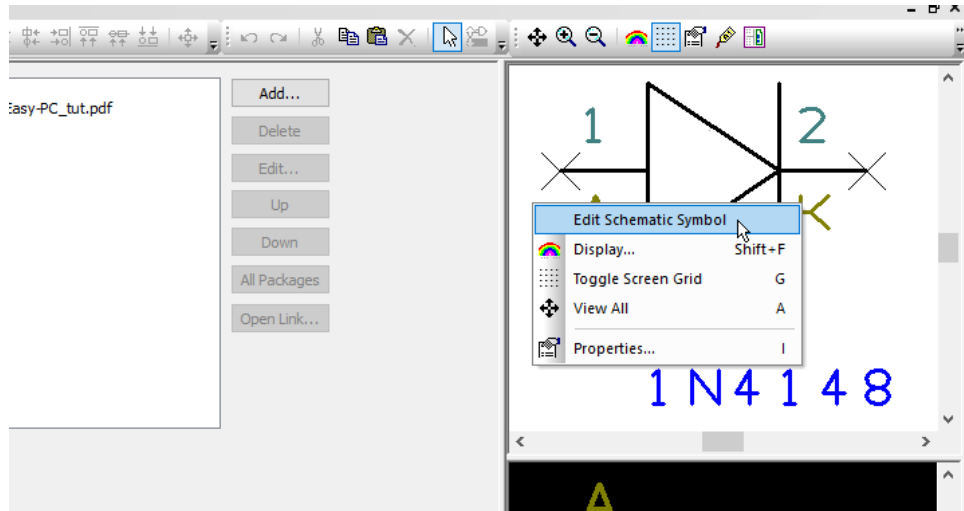
This option will recognise any value that starts with http, https, www., a drive name such as C: or a UNC path name such as \\server\path. If you select a value in the list whose text matches one of these patterns, then the **Open Link** button will become enabled. Clicking this button will pass the text string to the operating system to be opened in your web browser or relevant application directly from the dialog.

This **Open Link** button is also available on the **Values** tab of **Properties** for a **Component**, again allowing you quick access to reference material such as component datasheets directly from the application:



Refresh Symbol in Component Editor

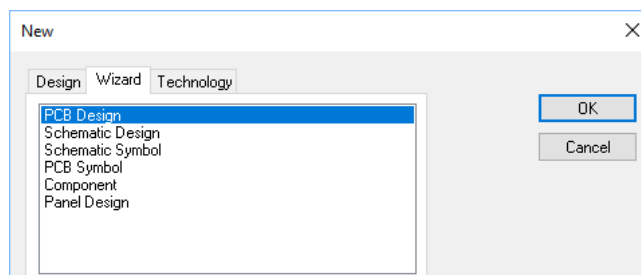
When you are editing a Component, you may find that you need to edit a Schematic or PCB symbol that is referenced by that Component. You can do this easily by right-clicking on the Symbol preview window and choosing **Edit Symbol in Library**.



When you have made the necessary changes to the symbol, saving it back to the library will now cause the previews in the Component to be refreshed by automatically reloading the newly saved symbol from the library. You no longer have to close and re-open the Component to see the changes.

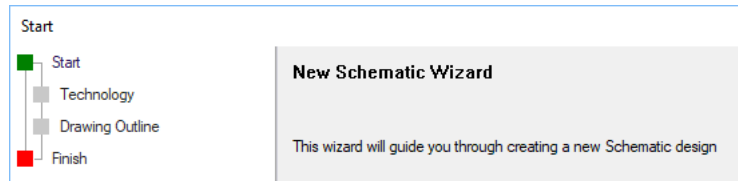
Wizards for New Schematic and PCB Designs

New Wizards are now available to assist you in creating new **Schematic** and **PCB** designs. These are accessed from the **Wizards** tab of the **New** dialog on the **File** menu:



New Schematic Wizard

The **Schematic Wizard** helps you to produce consistent schematic drawings by providing you with access to a **Technology file** to use and your favourite **drawing outlines**:



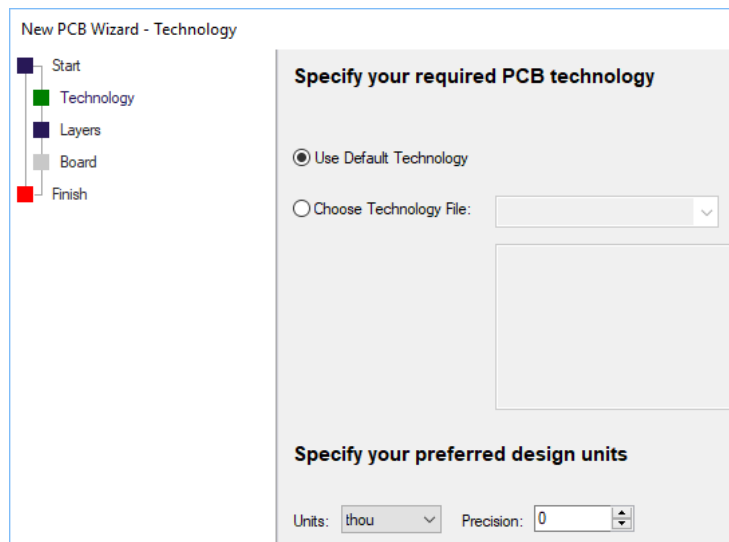
When you select **Finish**, you get the choice of writing the design to a given name and starting the design using that name (without saving it to file) or to write the design to file using a name that you specify and starting the new design.

New PCB Wizard

The **PCB Wizard** takes you through a series of steps that are intended to help you construct the underlying technology required for a particular PCB setup.

You can choose which **Technology** file to use, the **Units**, the **Layers** and the **Board Outline**.

Technology



The **Technology** page allows you to choose a technology file to use. You can either use the default technology set then add to it during the New PCB creation process in this wizard, or use your chosen Technology file. You can also your design **Units** and **Precision** to use.

Layers

New PCB Wizard - Layers

Start
Technology
Layers
Board
Finish

Specify which layers you require

Use Layers From Chosen Technology File
 Define Layers:

Electrical Layers
 Single Sided Board
 2 Layer Board
 4 Layer Board
 6 Layer Board
 8 Layer Board
 10 Layer Board
 12 Layer Board
 14 Layer Board

Powerplane Layers
 Layer 2 Using Net:
 Layer 3 Using Net:
 Layer 4 Using Net:
 Layer 5 Using Net:

Paste Mask for SMT
 Top Side
 Bottom Side

Solder Mask
 Top Side
 Bottom Side
 Cover Vias

Auto-Route Bias
 First Routing Direction:
 X
 Y

Allow Routes:
 On Top Side
 On Inner Layers
 On Bottom Side

The **Layers** page allows you to define the number of layers and associated layers for use in the design. If your chosen technology file on the previous page already has those defined, you can check the radio button to use the ones from the technology file. If you wish to craft them on this page, then make your selections.

Board Outline

New PCB Wizard - Board

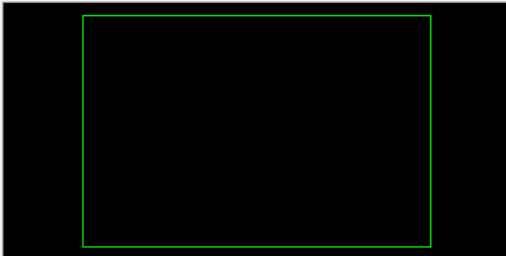
Start
Technology
Layers
Board
Finish

Define your board outline

Define Board Size:
 Circle Width: thou Height: thou

Use Board Outline From Chosen Technology File
 Copy Board Outline From Another File:

Board Preview:



The **Board** page enables you to define the board size or copy an existing board outline from another existing Easy-PC PCB design.

The **Board Preview** windows allows you to confirm the board outline chosen is the one required.

Finish

When you select **Finish**, you get the choice of writing the design to a given name and starting the design using that name (without saving it to file) or to write the design to file using a name that you specify and starting the new design.

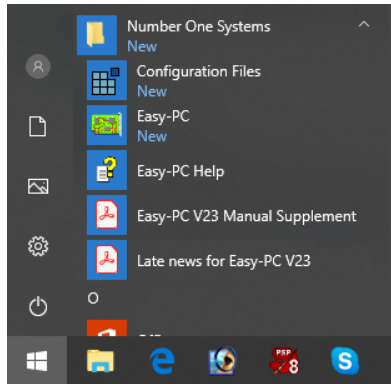
Configuration Files

To help you manage the transfer of application settings between computers, and to assist with the backup of application settings in case of system failure, you can now save and load Configuration Files using the new tool that is accessed from the **Configuration Files** command on the **Help | Support** menu.

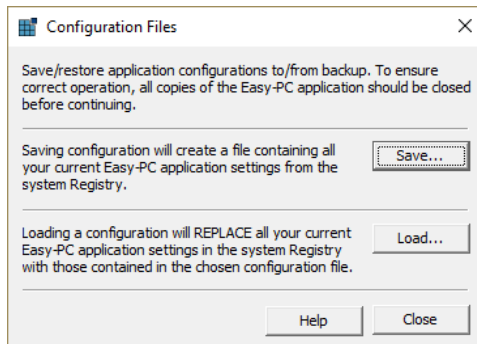
When you select this command, a separate application is launched to do the work. This application will save the existing settings from the 'current user' section of your system Registry to a file, or load a previously saved Configuration file into the Registry.

Management of Configuration files is done outside of the main application to ensure that the information in the Registry is fully up-to-date before a Configuration is saved, and that any information restored to the Registry is present for the Easy-PC application to load when it is next started up.

You should therefore close your Easy-PC application once the **Configuration Files** tool has been launched. You can also launch the tool from the shortcut on the **Start** menu that is placed there when the software is installed.



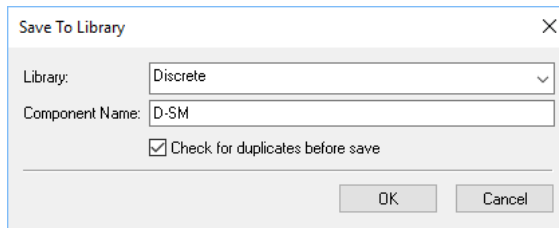
When run, you are presented with the **Configuration Files** dialog:



Configuration files contain a simple textual representation of the application settings. These are saved with the file extension .acf. We do not recommend that you attempt to edit these files; the application does not have any ‘special knowledge’ of the meaning of the various settings stored in the file, so alteration of the contents could lead to faulty application settings in the Registry which in turn could cause the main application to perform incorrectly or even be unstable.

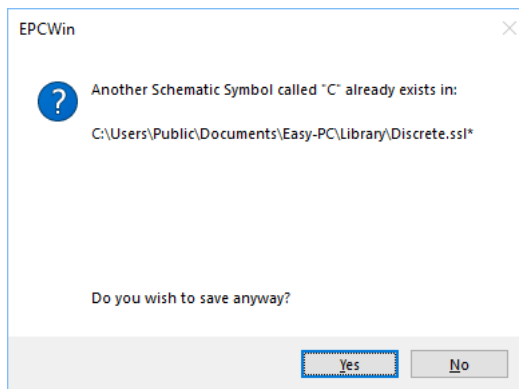
Save to Library Displays Information About Duplicates

There is a new check box on the **Save To Library** dialog within the **Library Manager** allowing you to check for duplicates. This operates on all library types; Components, SCM Symbols and PCB Footprints:



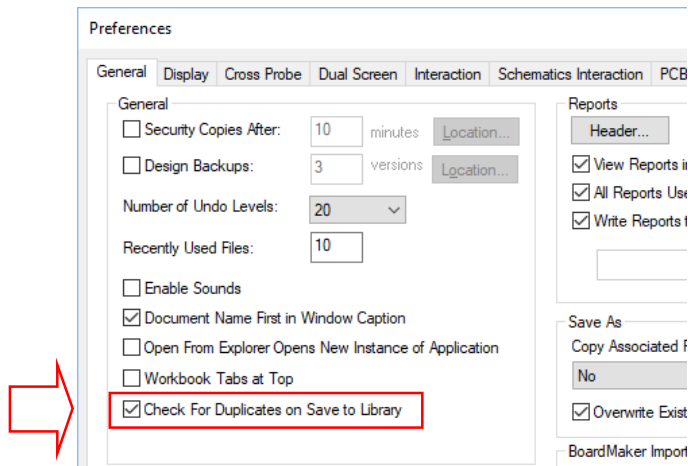
This will tell you if the library item you are about to save exists in any other currently enabled library, to help you avoid accidentally creating unwanted duplicate library items. You may still want to save the library item with that name, for example to create a custom copy of a symbol that is provided by the standard libraries so that your custom symbol is the one that is used when adding components to a design.

If any duplicate items are found, the names of the libraries in which they are located are listed on a popup dialog. If any of the libraries are earlier in the search order than the library to which you are about to save your item then these are marked with an asterisk to highlight the fact that these would be used in preference to the one you are attempting to save.



The same check is also available in the **Save Items to Library** and **Import Items to Library** dialogs available on right-click from design and drag-and-drop (or **Add File**) in **Library Manager**. As these dialogs handle multiple library items, those items that have duplicates will be marked but there won't be a list of library file names as each component or symbol could be duplicated in different ways. However, this should still provide you with sufficient information to determine the possible effects of having those duplicate items present in your set of library files.

You can also alter this setting with the check box on the **Preferences | General** dialog:



Add Component and Library Manager Dialogs Resizable

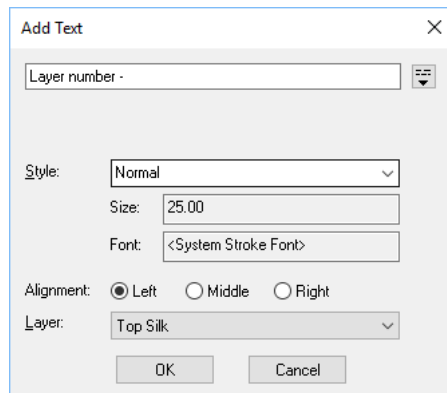
For more flexibility in displaying long component and path names, the dialogs for **Add Component** and **Libraries** can now be resized. To resize the dialog, grab the corner or bottom or right-hand edge of the dialog using the cursor to resize it.

Export Images as Bitmap and Metafile

This feature gives us two new commands on **File** menu **Export >** to output image to **Bitmap** and image to **Metafile**.

Add Text Choose Style and Alignment

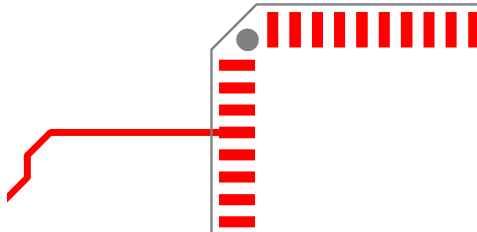
When adding new text items to a design or symbol, controls are now available on the **Add Text** dialog to allow you to choose the **Text Style** from the currently defined list of Text Styles in **Design Technology**, and to adjust the text **Alignment**, all before placing the text. Previously you could only alter these by separate commands either during placement of the text or in **Properties** after the text has been placed. Properties can still be used to change the Style and Alignment of course.



Add Track or Shape Off-grid

When adding or editing a track or shape, holding the **Ctrl** key down will temporarily disable gridding, to allow you to place the next corner off the grid.

This could be a useful feature if, for example, you want to snap a track into a fine pitched SMD pad but want the routing to stay on a fairly coarse grid:

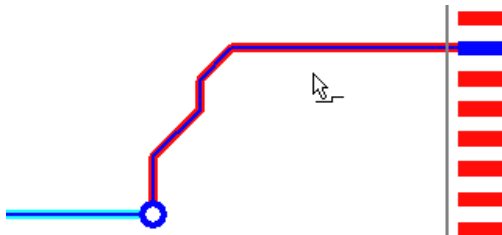


Note: We advise that you use the function sparingly to avoid shapes that are off-grid.

Highlight Net Interactive Mode

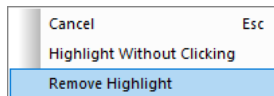
There is a new command on the **View** menu under **Highlight Net** named **Interactive**. This provides a special ‘mode’ of selection which highlights nets as items are selected.

Whilst in this mode, you can click on any item in the design and if that item belongs to a net then the whole net will be highlighted (drawn in the **Net Highlight** colour). Select a different item and the first net is redrawn without highlight and the newly selected net is then highlighted.



A special cursor confirms you are using this mode.

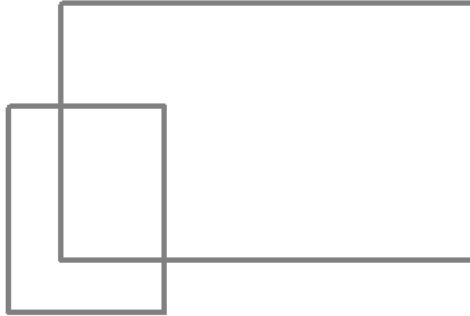
On the context menu in this mode you can toggle **Highlight without click** which flips between normal select mode (pick as you click) and an “auto select” mode where it will continuously attempt to pick items as you move the cursor, providing you with instant feedback on the connectivity in your design.



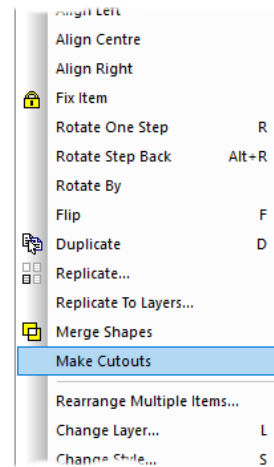
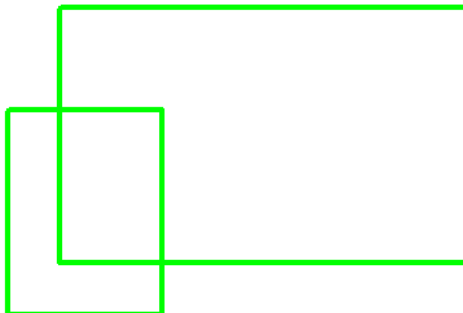
From the context menu, you can also **Cancel** this mode and use the menu to **Remove Highlight**.

Make Cutouts from Shapes

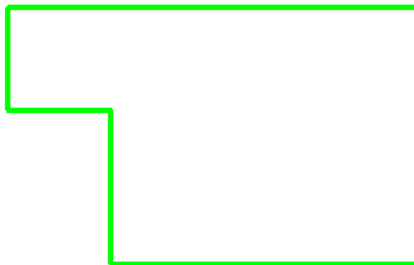
The new command, **Make Cutouts**, appears on the shortcut menu in **Select Mode** when you have two or more **closed** shapes of the **same type** (and on the **same Layer** in a PCB or footprint) selected.



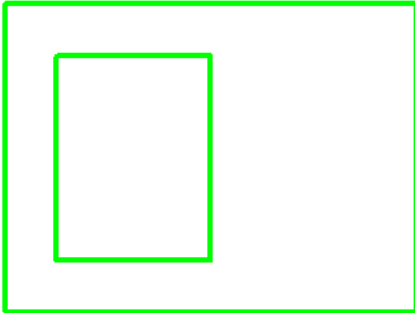
The **selected** shape with the largest area is used as the 'parent' shape, and all the other selected closed shapes that overlap with this parent shape are used to make cutouts of the outer shape. The original 'inner' shapes are then deleted.



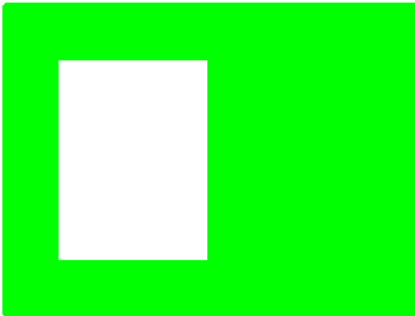
Choosing **Make Cutouts** on the context menu, produces this below. The smaller shape has been 'cut' from the larger shape.



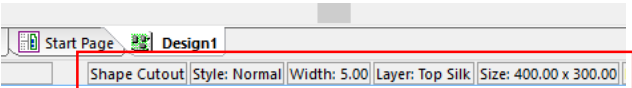
Another example of this would be to use the second shape to add cutout detail to the parent shape:



Selecting both shapes will cut the smaller shape from the larger one, evident once the shape is filled:



Confirmation that a shape is a cutout can be seen on the **Status bar** with the cutout shape selected:



This command can be useful for example if you have imported DXF data to create a company logo but it is only defined as a set of nested shapes without any information about cutouts or filling.

For example, you can take this set of five separate shapes (two nested inside larger shapes) as might be the case for text shapes:

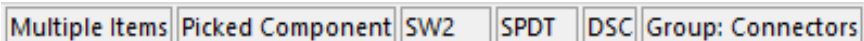


Using this command, quickly turn it into these three filled shapes:

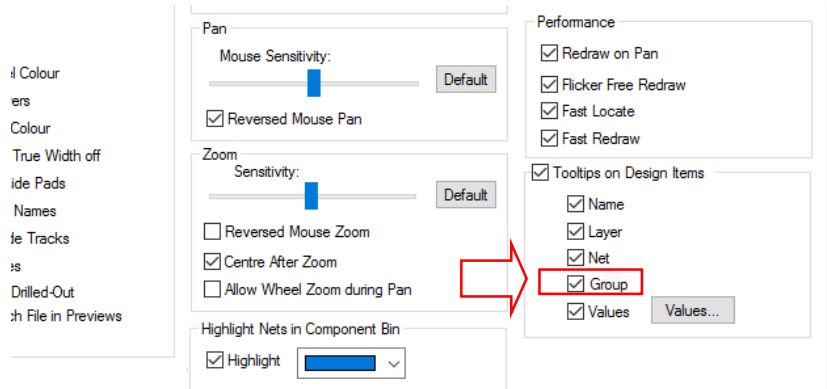


Group Name Shown On Status Bar

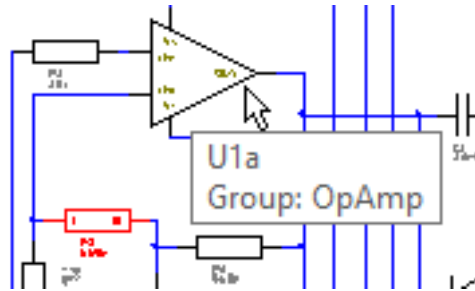
The **Status bar** will now show the name of the **Group** to which the selected item belongs:



There is also a setting on the **Preferences | Display** dialog to include the **Group** name on **Design Tooltips**:



When viewed in the design, with one or more items within the group selected, the tooltip will appear like this:

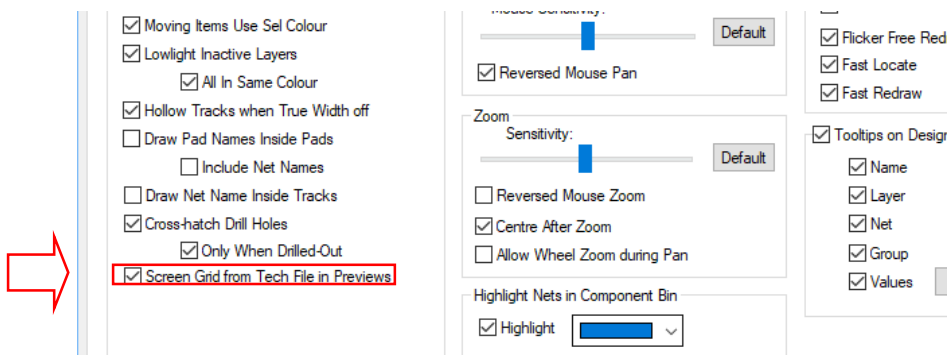


Display Grid in Previews

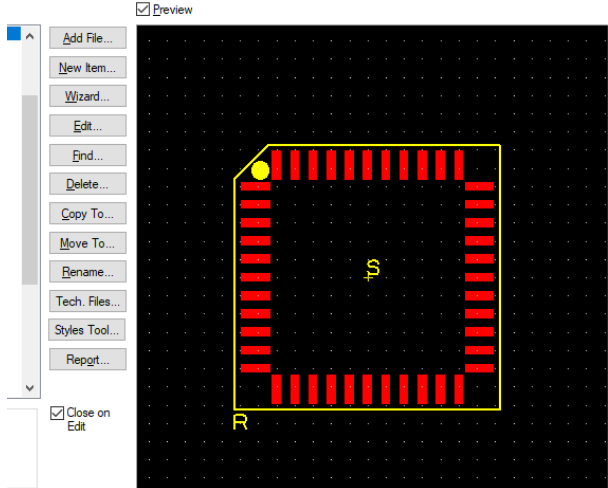
There is a new check box in **Preferences | Display**, called **Screen Grid from Tech File in Previews**.

With this selected it allows you to override the normal behaviour of suppressing any visible screen grid in symbol previews. If you check the box, any visible grid in the current technology file of the relevant type will be drawn in the preview.

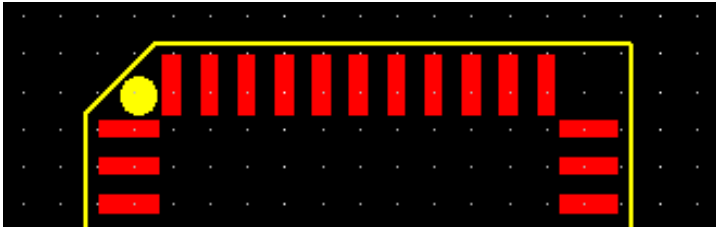
In summary, checked will suppress the screen grids in symbol previews. not checked will draw the grids.



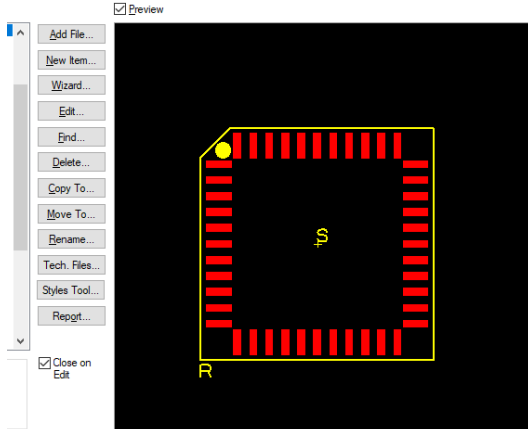
With the check box checked:



The grid dots are quite subtle, this is what a zoomed in view looks like:

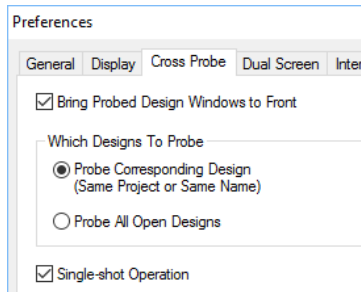


With the check box unchecked grid dots are always forced off:



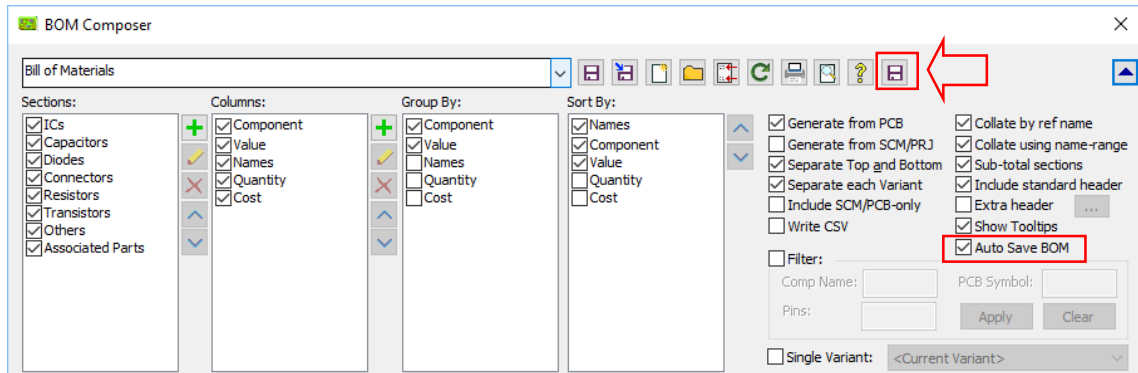
Cross Probe Single-shot Operation

A new check box on the **Preferences** dialog and **Cross Probe** page has been added. This new feature enables **Single-shot Operation**. This will tell the Cross Probe mode to probe once and then return to Select Mode. If this box is unchecked, then you will stay in Cross Probe until you cancel out or select some other mode.



BOM Composer turn off Auto-generate Report

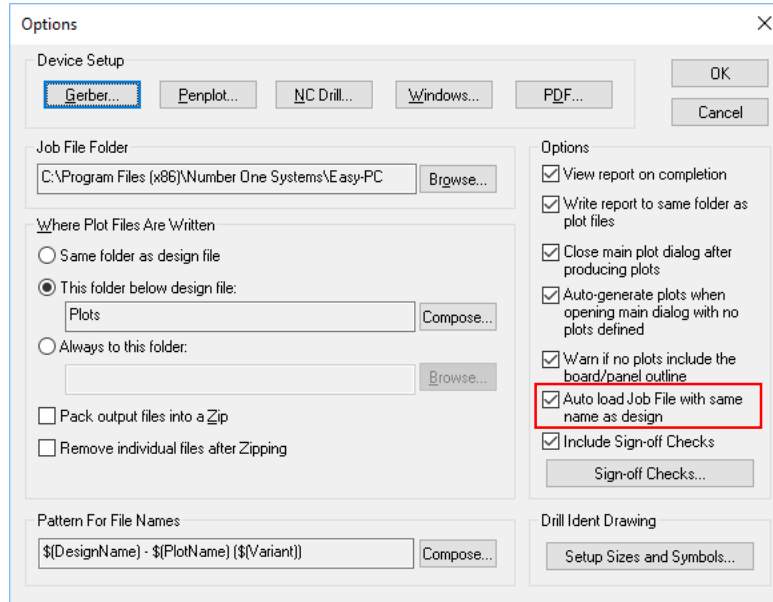
The normal operation of **BOM Composer** is to generate the report with every alteration you make to the report template or settings so that you can quickly adjust the operation.



With the BOM options unfolded, the new **Auto Save** check box allows you to override this behaviour, and only generate the report when you check the new **Save** button. The on-screen report will still be updated as you make changes to settings or the report template, but the files needed to update the on-screen display are generated in a temporary folder so that they do not affect your own project data.

Plotting Auto-load Job File

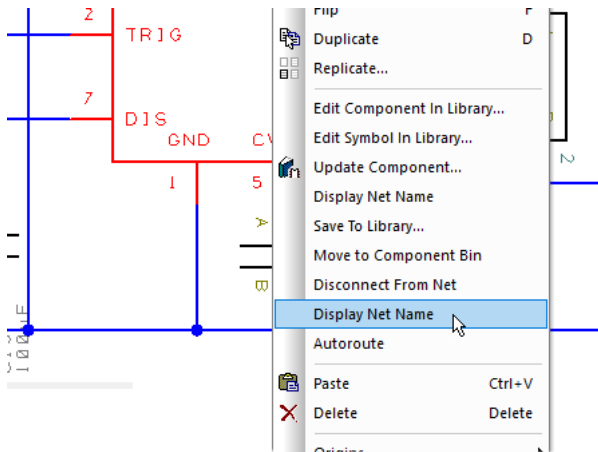
On the **Plotting and Printing** dialog and **Options**, there is a new check box **Auto load Job File**. With this checked, opening the Plotting dialog will automatically load a job file from the same location as the design. For a PCB design or a SCM that is not in a project, it will look for <designname>.EPJ, for a SCM in a project it will look for <projectname>.EPJ.



Display Net Name on Component Select (SCM)

You can now select a whole **component** and show/hide net names for all connected pins/pads on that component in one go. Previously you could only do this for a single pad.

When you have multiple items selected and this includes one or more components, the **Display Net Name** command will show the net name for all connected pins on those components.

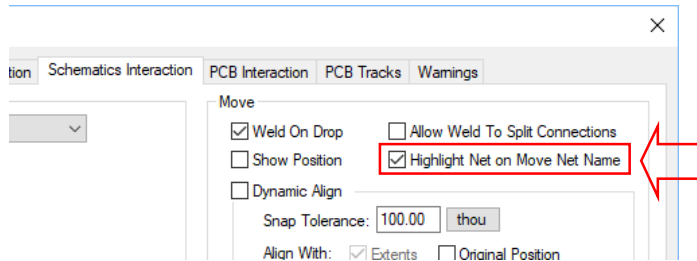


You can also switch off all net names associated with the component by selecting it and unchecking **Display Net Names** from the context menu.

Move Net Name Highlights Net (SCM)

In a Schematic design, when you **move a net name** you now have the option to automatically **highlight the entire net** to which that net name belongs.

This option is controlled by a new check box in the **Preferences dialog** under **Schematics Interaction** called **Highlight Net on Move Net Name**.



Dangling Connections Report (SCM)

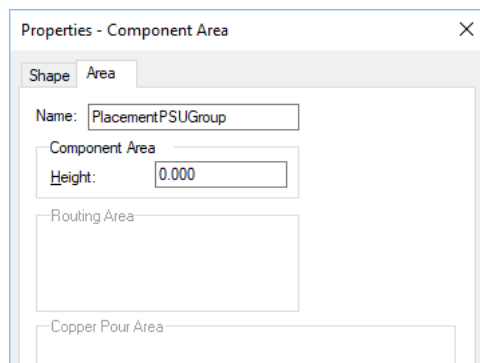
Within the **Schematic design editor**, the **Dangling Connections** report is available from the **Output** menu, **Reports**. This will show you any connection that doesn't end on either an electrical pin or doesn't connect to another point using an implied net name

Height Check Areas Renamed to Component Areas (PCB)

Height Check Areas have been renamed to **Component Areas** to reflect a wider range of their use, such as for use with Component Rules (see below) as well as their height checking capabilities.

Component Areas

When editing the **Properties** of a **Component Area**, you can assign it a name but with no height value. If you add a Height value, it can be used for multiple uses.

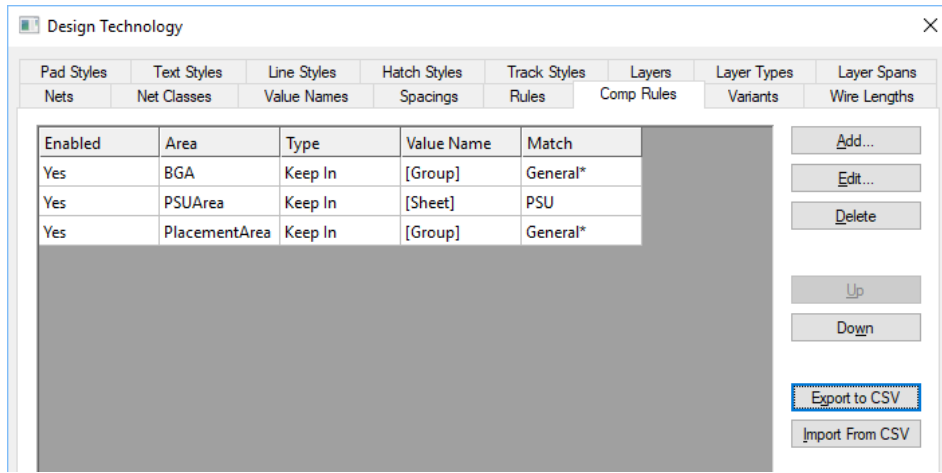


This can then be used within the **Component Rules** and **Auto-Place** feature.

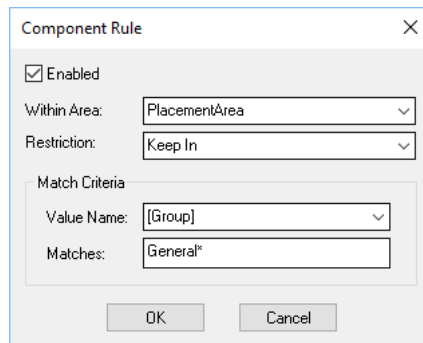
Component Rules (PCB)

Component Rules allow you manage your component placement using simple keep in/out rules that use wildcard matching of **Component Properties** to areas. (Height Check areas have been renamed to Component Areas to reflect this additional use).

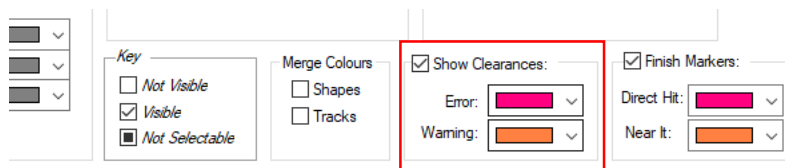
Component Rules are set up on the new **Comp Rules** page on the **Design Technology** dialog.



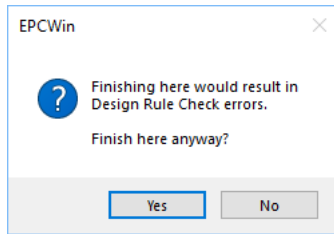
Each rule uses a wildcard string to match the desired components by **value**, **reference name**, **component name**, **group** or **sheet** name or within a **named** area.



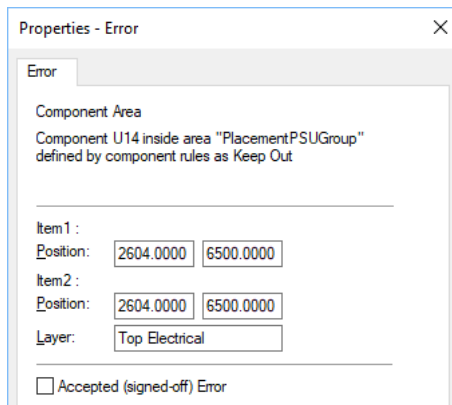
When moving a single component, areas which are relevant to the rules for that component are drawn highlighted in two colours: in the **normal Highlight** colour for **Keep In** areas, and in the **Clearance Error** colour for **Keep Out** areas where that component should not be placed. This includes highlighting as 'keep out' any areas which specify a height restriction lower than the height of the component being placed. These visual cues provide you with instant feedback to assist with your component placement.



With the **Online DRC** option enabled, you also get an instant warning

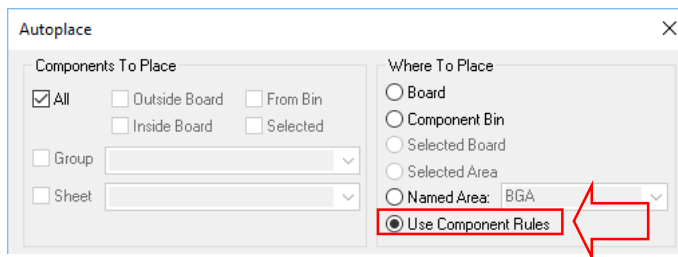


Overriding this DRC violation will write a **DRC Error Marker**, **Properties** of this marker will show you the error:



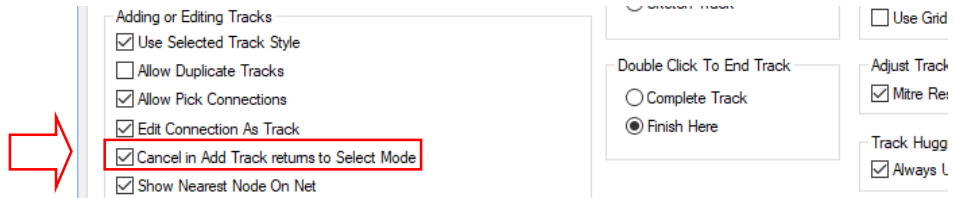
Auto-place Using Rules (PCB)

As well as the **Component Rules** assisting with manual placement, the **Auto-place** dialog has a new option to make use of these rules when placing components.



Add Track Action on Cancel (PCB)

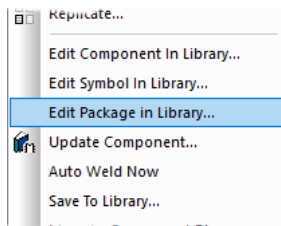
There is a new check box in **Preferences, PCB Tracks**, for **Cancel in Add Track returns to Select Mode**. The default setting is True (to follow existing behaviour).



If this box is checked, then pressing Escape or cancelling while adding a track will cancel the mode and return to **Select Mode**. If instead this box is not checked, cancelling the current track will stay in **Add Track** mode waiting for you to start the next track.

Edit Package in Library (PCB)

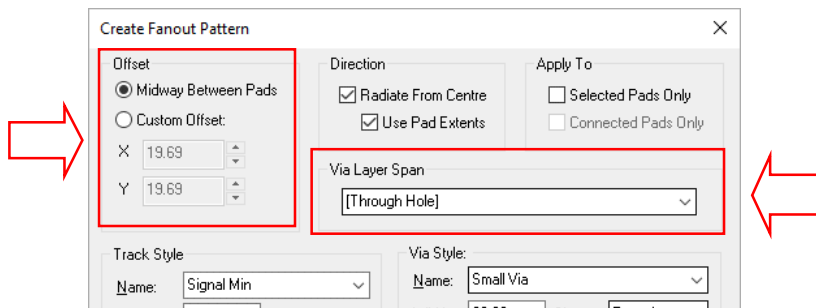
If you right-click on a component, you can now select a new command **Edit Package In Library**. This will open the **3D package edit** dialog showing the 3D package that corresponds to the component's footprint.



Enhanced Functionality for Create Fanout Pattern (PCB)

The **Create Fanout Pattern** dialog on the **Tools** menu, **Auto Fanout** > dialog has been enhanced. It now has controls to define how to automatically calculate the offsets so that vias are placed midway between rows/columns of pads (previously the values were calculated based on the extents of pad and via).

You can also choose the **Via Layer Span** from the drop-down list of spans to be used for the vias (previously always inserted them using Through-Hole span).



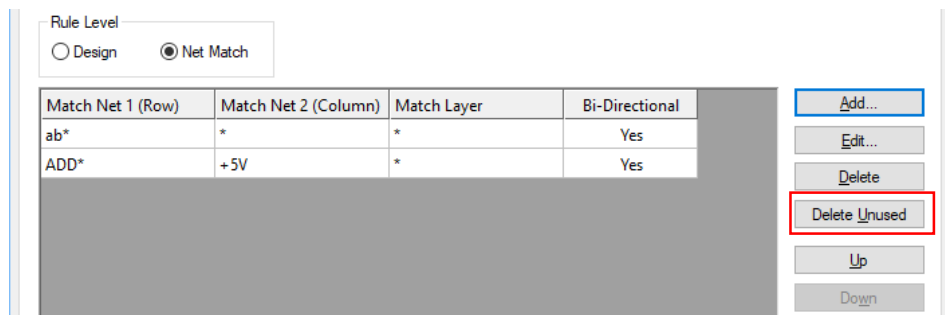
Online Design Rule Check for Components (PCB)

When using the **Online DRC**, the functionality has been enhanced to include **Component Area** and **Component Height** checks when placing a component. This ensures any rules set up through the Component Rules dialog or Component Height Properties will be checked dynamically.

Delete Unused Net Match (PCB)

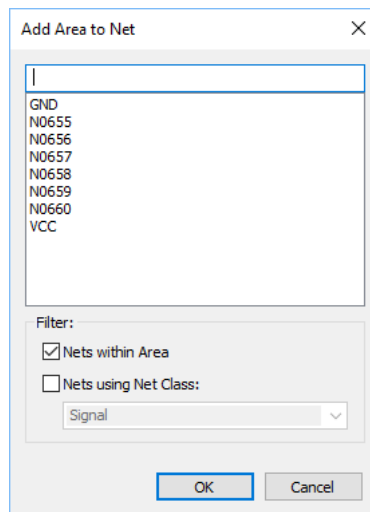
To assist in ‘trimming down’ a set of **Net Match** rules defined in a technology file to retain only those that apply in the current design, a new button, **Delete Unused**, is now available on the **Spacings** page of the **Design Technology** dialog.

When you click this button, all the currently defined Net Match rules will be checked against the nets and net classes defined in the design, and any rules that do not match with anything will be deleted.



Add to Net on Pour Area (PCB)

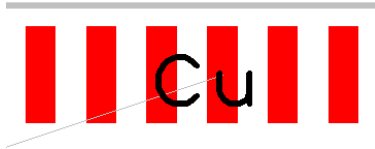
When you right-click on a **Copper Pour Area** that is not yet assigned to a net, choosing the **Add To Net** command will take you to a new dialog where you can use ‘filter’ options to help you quickly find the correct net for your Area.



The list will contain net names inherited from the design where the area overlaps items on those nets. If the area does not cross anything, it will have no net names. In this instance you can type the net name required to add it to that net, or type a new net name to create a new net for this area.

Pour Copper adds DRC Error Markers (PCB)

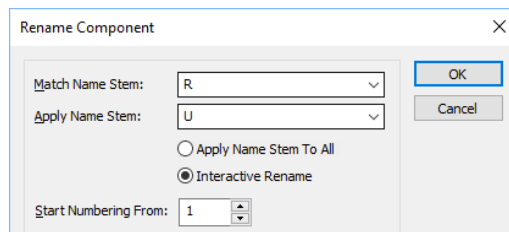
When using the **Pour** option, if it finds any issues it needs to tell you about (e.g. insufficient thermal spokes, copper area too small or unconnected) these are already listed in the report, but it will now also add **DRC error markers** (code “Cu”) to the design so that you can find them using the **Goto** bar.



When you re-pour or clear any copper area, any copper DRC markers within that area and on the same layer as the area are removed.

Rename Component “immediate” mode (PCB)

The existing point and click interactive **Rename Components** mode on the **Tools** menu has been extended to allow a simple “immediate” rename of all components matching the supplied name stem.



On use, the option returns to the **Rename Components** dialog ready for the next action.

Also, there are additional radio buttons on the dialog.

Apply Name Stem To All

When using this mode (which is only available if the two component name stems are different) then the rename action will take place as soon as you click the **OK** button.

A message will be displayed showing how many components were renamed, and you will be able to choose whether to return to the dialog to rename some more components, or exit from the rename operation.

Interactive Rename

When using the Interactive Rename you need to enter the start number from which the 'new' component names should begin.

When you click **OK**, the dialog closes and the 'point and click' renaming can begin. All the components matching the chosen stem will be highlighted to help you see them easily on your design.

Each time you select a component, the current name in the sequence is assigned to that component, and the sequence number is incremented. The next name to be applied is always displayed on the Status Bar.

If the next name to be applied is already in use elsewhere in the design, the application will ensure that the names are swapped over so that no duplicate names exist.

Swap Components (PCB)

The ability to swap two components to exchange their positions has been added.

Select the option from the **Edit** menu and **Swap Component Positions**, select the first component and then the component to swap with.

You can also select the two components first, then choose the option from the **Edit** menu.

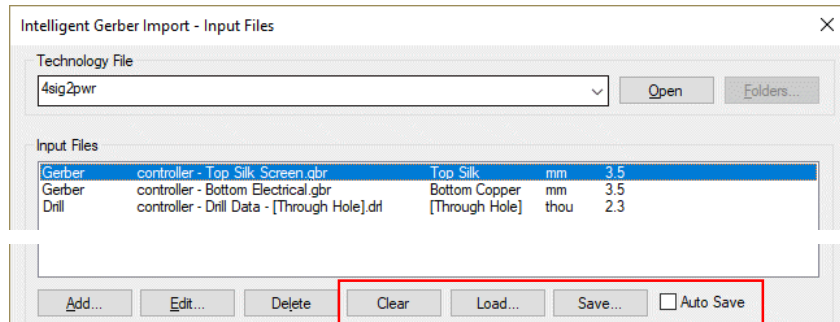
This option will swap similar components and components that are unlike, provided they can be swapped, for example if it creates a DRC error because of a Component Rules area violation.

Intelligent Gerber Import Improvements (PCB)

Drag and Drop

You can now take Gerber or Excellon or BOM files and drag them from your desktop or Windows Explorer window and drop them onto the Intelligent Gerber Import dialog. This will operate the same as if you had used the Add button on the dialog but is obviously much quicker.

New Dialog Navigation



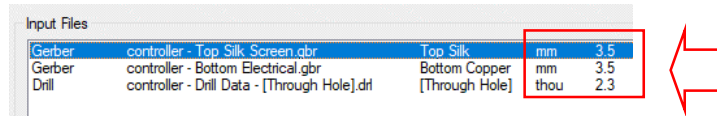
There is a new **Clear** button to empty the list of input files.

There are new **Load** and **Save** buttons to load and save settings files that define a set of files to import.

There is a new check box option to **Auto Save** (and auto load) the dialog settings so that the next time the dialog is opened, it returns with same files as before.

More File Information

There is more information about file contents displayed on the dialog. For example, the file Units and Gerber format.



Better Layer Name Matching

The Intelligent Gerber Import dialog now does a much better attempt at name matching of Gerber files when matching the filename to Layers in the design, such as Silkscreen, Solder Resist, etc.